

Asian Nuclear Prospects 2012 (ANUP2012)

Design Optimization of Gas Separator in High Temperature Gas Cooled Reactor through Numerical Simulation Method

WANG Jin-hua*, TANG Yong, HUANG Yi-fan

Key Laboratory of Advanced Reactor Engineering and Safety of the Ministry of Education, Institute of Nuclear and New Energy Technology, Tsinghua University, Beijing 100084, China

Abstract

The gas separator is the key apparatus of the fuel handling and storage system in High Temperature Gas Cooled Reactor (HTGR), the flow field and pressure field in the gas separator with different design schemes were studied with computational fluid dynamics program FLUENT, which could provide theoretical evidence for design optimization of the gas separator. The result showed that impinging jet was detected in the initial design scheme, which caused vortex generation, besides, the interaction between impinging jet led to vortex between small holes. The size and number of the small holes in the horizontal pipe wall had significant effect on reducing the flow resistance. The pressure drop of the gas separator could decrease significantly by increasing the holes number under the vertical pipe as much as possible, which could reduce the mix between the jet and the surface. The pressure drop in the final design fell to about half of the value compared with the initial design scheme, which could reduce the energy consumption, and improve the system operation's stability.

© 2013 The Authors. Published by Elsevier Ltd. Open access under [CC BY-NC-ND license](#).

Selection and peer-review under responsibility of Institute of Nuclear and New Energy Technology, Tsinghua University

Keyword: High Temperature Gas Cooled Reactor; Gas Separator; Numerical Simulation; impinging jet

* Corresponding author. Tel.: 010-62788595-836; fax: 010-62795146.
E-mail address: wangjinhua@tsinghua.edu.cn.

1. Introduction

10MW high temperature gas cooled reactor use the spherical element as fuel, which could recycle continuously in the reactor core. The fuel's nominal diameter is 60mm, and the fuel's weight is about 0.2 kg. The fuel element's burn-up is measured after discharged from the reactor core one by one with γ radiation detect method, if it has not reached the final burn-up, the fuel element would be reloaded back to the reactor core, otherwise, it would be unloaded to the spent fuel canister for interim storage. The damaged fuel elements and debris would be loaded to the shielded fuel tank.

The fuel element could be lifted in the vertical pipe line pneumatically, the driving gas enter the fuel element transfer pipe at the bottom through a tee pipe coupling, and discharged at the top of the pipe through a gas separator, which is an important component in the fuel element transfer pipe system.

The function of the gas separator researched in this paper is separating the fuel element and the gas in pipeline. The flow field in the gas separator was studied through CFD simulations, and the mechanism of the pressure drop was analyzed in the paper, then the structure design of the gas separator could be optimized through CFD simulations in order to reduce the pressure drop.

2. Numerical methods and turbulence models

The CFD program FLUENT was used to simulate the internal flow field of the gas separator. The simulation model was constructed based on the design scheme of the gas separator, the inner diameter of the fuel transport pipe with holes is 65mm, as shown in figure 1. In the horizontal pipe, the gas drives the fuel element transport at the front of the small holes, as there's a single apparatus at the downstream of the gas separator, the single apparatus permits the fuel element pass through when revolving like a revolving door. After passing through the gas separator, the fuel element transports forward and the gas separates from the gas separator and flows to the vertical pipe at the top of the gas separator. The tetrahedral mesh model was created in the ICEM CFD program; and the grid was refined near the wall and hole regions.

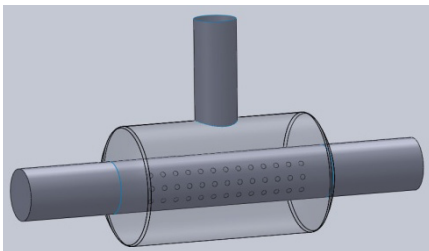


Fig. 1. (a) Three dimensional model of the gas separator

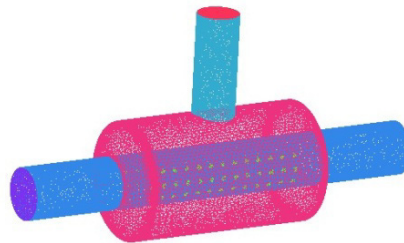


Fig. 1. (b) Mesh model of the gas separator

In the FLUENT program, the calculation model and parameters were set up due to the flow condition, the gas type is set as air in atmosphere temperature and pressure, the realizable $k-\epsilon$ model was set as the turbulence model, and the velocity inlet and the pressure outlet were set as the boundary conditions. The wall was treated as the non-slip wall, the second order upwind scheme was used to solve the scattered equations, and the semi-implicit method pressure linked equation (SIMPLE) was used to solve the control equations. The working pressure is set as the standard pressure, and the temperature is 293.15K. The working gas is set as the incompressible fluid.

In order to optimize the structure design of the gas separator to reduce the flow resistance of the gas separator, numerical simulation of the gas separator's internal flow was conducted, and the mechanism of the flow resistance were analyzed.

3. Results and analysis

3.1. The static pressure drop

The pressure drop in different inlet flow rate was calculated with the gas separator, and the results were shown in figure 2, it is shown that in the calculation of initial design scheme, the static average pressure drop of the import and export section increase with the increase of inlet velocity.

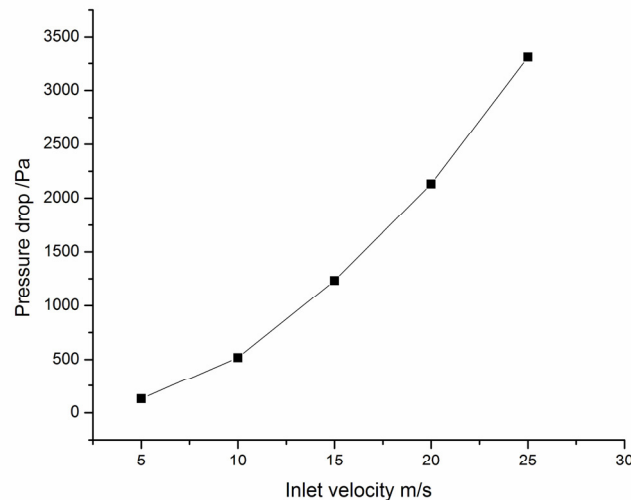


Fig. 2. Static pressure drop of initial design under different inlet flow rate

The cross section of the velocity vector field at the middle position ($x=0$) of the initial design is shown in figure 3. As the area of the holes on the inner wall is small relative to the radial size of the gas separator, several jets produced from the holes experience the freedom jet region, and then hit the outer wall surface after the collision region, and form the wall jet region near the outer wall surface, which resulted the pressure drop of internal flow. The wall jets of the bottom holes encounter in the bottom of the vessel, which formed whirlpool, the whirlpool distributed symmetrically on both sides, flowing upward along the horizontal pipe wall.

The fluid between the holes in the axial direction flowed into the free jet region due to entrainment of the flow from the holes, and then mixed with the jet near the holes, and then vortex was formed between the holes, which resulted in the pressure drop. The gas entered the vessel through the holes, the flow jet from the lower holes form an impinging jet in process of upward flow with the jet from the upper holes, which lead to the formation of scarf vortex, horseshoe vortex, wake vortex, etc.

Two adjacent holes constitute double holes impinging jet, and form the upper jet between the holes, as shown in figure 3. The whirlpool area was formed between the upper jet and the jet flow. Due to the limitation of the gas separator's size, the jet flow inside the vessel is inevitable, which result in the formation of the vortex and pressure drop of the internal flow.

Based on the above analysis, it was considered to adjust the arrangement of holes through changing

the position to reduce the pressure drop. The result showed that the arrangement of holes has little effect on the pressure drop due the jet flow's high velocity relative to the size of the gas separator, the gas flowing out of the holes encounters the vessel wall directly, there's a lot of swirl in the gas separator's vessel, which result in the little effect to decrease the pressure drop by changing the holes distribution.

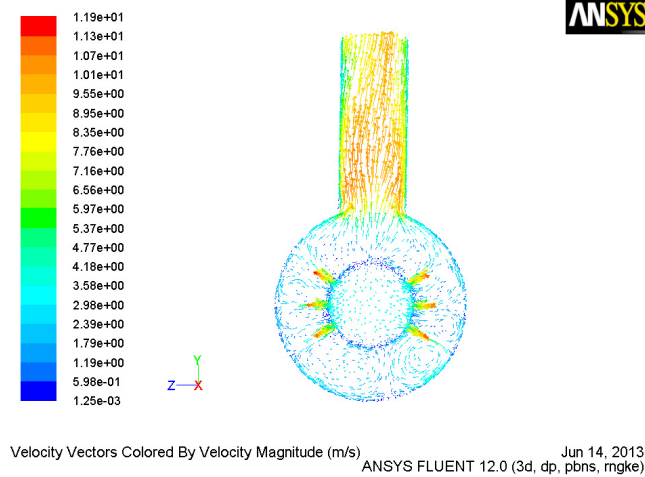


Fig. 3. Velocity vector at the section X=0 of the gas separator

3.2. Effect of the hole dimensions

It is required to reduce the mix of the jet with the surrounding fluid in order to reduce pressure drop of the gas separator, the method of introducing free jet is useful to avoid the formation of the vortex. In order to decrease the pressure drop of the gas separator, the flow field of several designs of the gas separator were calculated, the results showed that, the method of increasing the size of holes and increasing the number of holes has significant effect on decreasing the pressure drop of the gas separator. The reason is that if we increase the hole size, the flow velocity flow out of the holes would decrease, and the vortex formed between jet and wall would decrease, which lead to the decrease of the pressure drop.

To analyze the sensitivity of the hole dimension to the static pressure drop of the gas separator, the pressure drop of several gas separators with different radius were calculated, and the relationship between the static pressure drop and the hole radius was obtained, as shown in figure 4, which showed that, it is useful to reduce the pressure drop of the gas separator through increasing the holes radius. The radius of the hole was chosen as 4 mm finally.

3.3 The pressure drop of the contraction

There is significant local pressure drop at the vessel wall and the vertical pipe joint. The flow velocity increases at the contraction region, which cause a pressure drop. Therefore, in the optimized design scheme, it is considered to make a chamfer between the vertical pipe and the vessel to reduce the pressure drop at the contraction region.

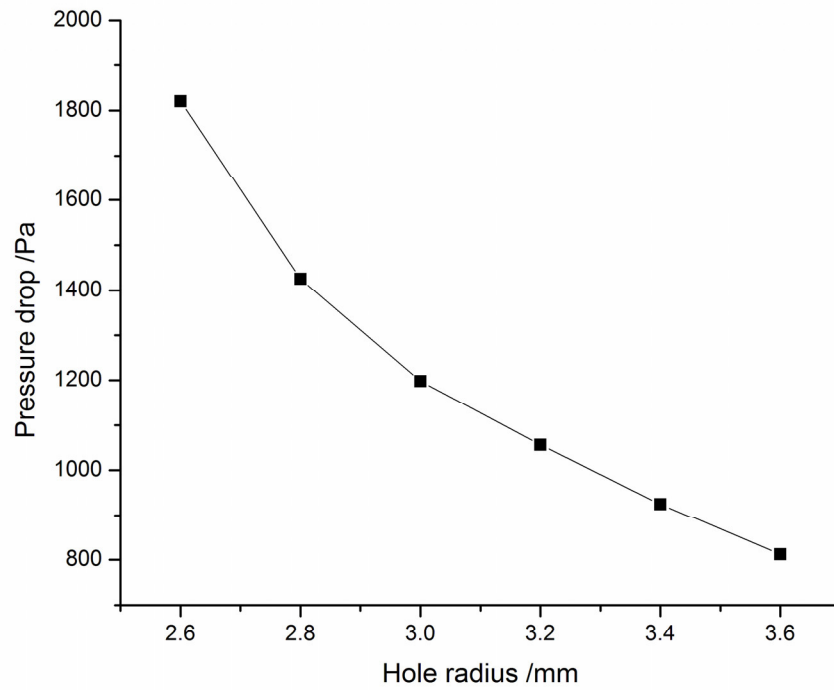
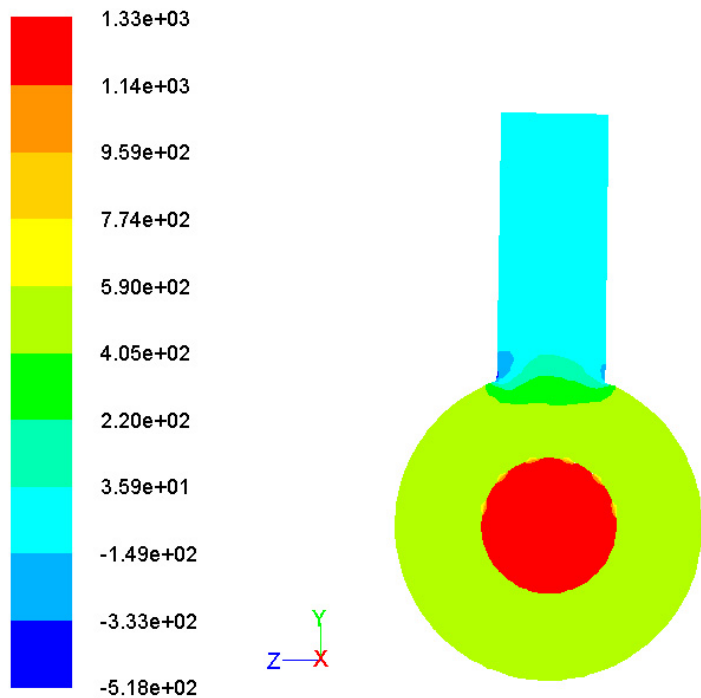


Fig. 4. Relationship between the static pressure drop and the radius



Contours of Static Pressure (pascal)

Jun 15, 2013
ANSYS FLUENT 12.0 (3d, dp, pbns, rngke)

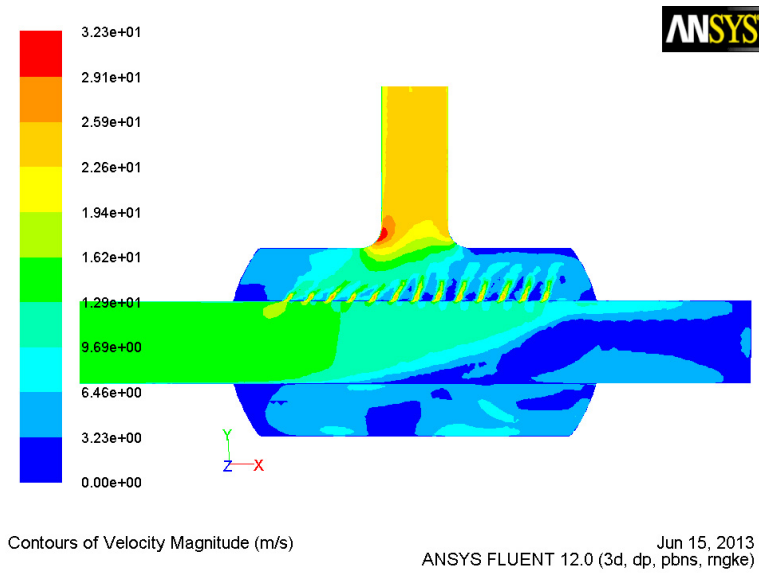
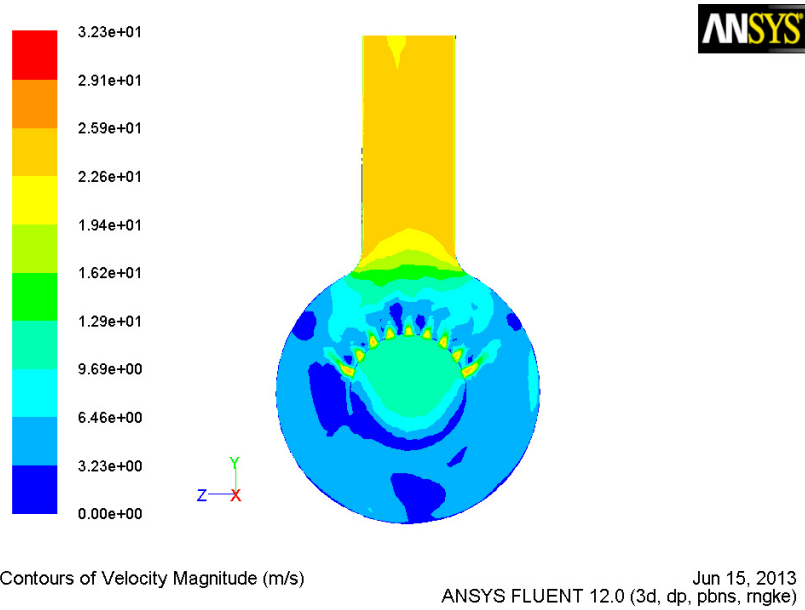
Fig. 5. Static pressure cloud map at the section X=0

3.4 Results of the optimization

The optimization method of the gas separator could be summarized into two aspects: the first is to increase the size of the holes and increase the holes number, and set holes under the vertical pipe as much as possible; the second is to make a chamfer between the outer wall and vertical wall.

Compared with the original design scheme, 9 rows holes are arranged on the small wall in radial direction, and there are 13 holes in each row. The holes under the vertical wall could form free jet gas flow. The holes are symmetrically arranged at two sides of the pipe wall. The pressure drop of the gas separator fell to about 50% of the initial design. Optimized velocity and pressure flow fields are shown in figures 6 and 7.

The mass flow distribution of the 13 holes at the top of the pipe is shown in figure 8. The mass flow increases from hole number 1 to 8, decrease from 8 to 12, and again increase at 13, the flow distribution's reason is a combination of the gas flow's inertia and the hole's distance from the vertical flow pipe.

Fig. 6. (a) Cloud map of Velocity at the section $Z=0$ of the final designFig. 6. (b) Cloud map of velocity at the section $X=0$ of the final design

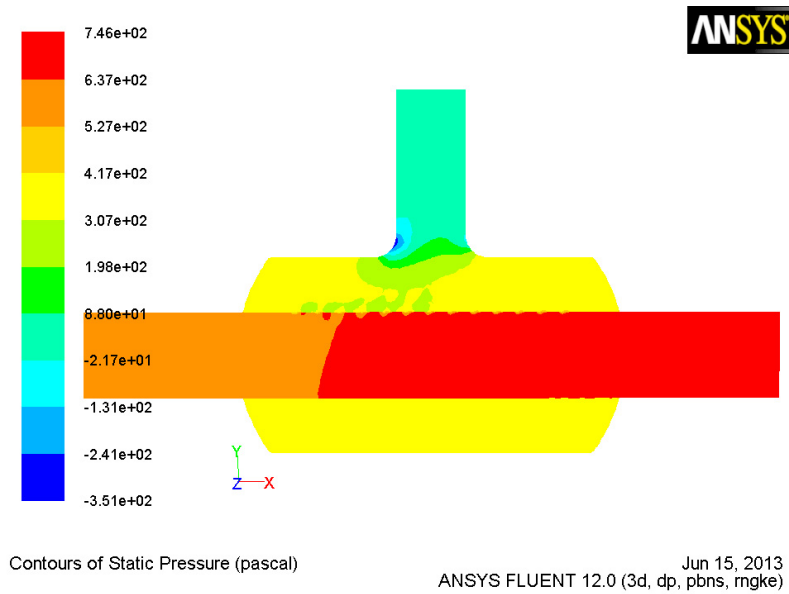


Fig. 7. Cloud map of static pressure drop at the section Z=0 of the final design

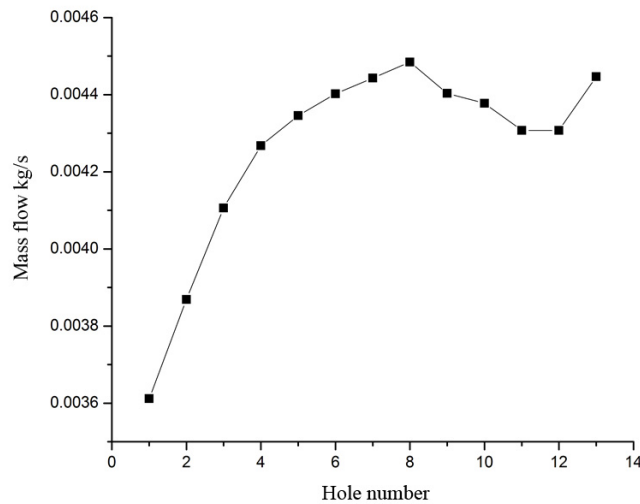


Fig. 8. Flow distributions of the 13 holes at the top of the pipe

4. Conclusions

1. The flow resistance of the gas separator comes from the flow region cross-section's mutation and the internal jet vortex mainly.
2. The pressure drop of the gas separator could decrease significantly by increasing the number and size of the holes, and the holes size affect the pressure drop significantly.
3. The pressure drop of the gas separator could decrease significantly by increasing the holes number under the vertical pipe as much as possible, which could reduce the mix between the jet and the surface.

4. In the final optimized design scheme, the pressure drop of the gas separator between the inlet and the outlet is about 50% of the initial design.

Acknowledgements

The author acknowledges the sustentative fund of major national science and technology projects (2012ZX06901021) provided by the National Energy Bureau of China.

References

- [1] Wu Hailing, Chen Tingkuan, Luo Yushan. Numerical Simulation of 2D Jet-to-Crossflow Heat Transfer with Different Turbulence Models. Journal of Xi'an Jiaotong University; Vol.35 No9 2001.
- [2] Zhang Yan. Experimental and Numerical Investigations on the Vortical Structures of an Impinging Jet in Crossflow. 3rd ed. New York: Macmillan; Shang Hai University, Shang Hai 2005.
- [3] Tong BingGang, Yin XieYuan, Zhu KeQing. Theory of vortex kinematics [M]. He Fei, Press of Chinese University of Science and Technology. 1994. 13-17.
- [4] Yu Changshao. Turbulent jet current [M]. Beijing, Higher Education Press. 1993. 149-155.
- [5] Dong Zhiyong. Impact jet flow [M]. Beijing, Ocean Press. 1997. 176-180.